19940020874

N94-25356

UNIFIED APPROACH FOR INCOMPRESSIBLE FLOWS

Final Report
NASA/ASEE Summer Faculty Fellowship Program-1993
Johnson Space Center

Prepared by:	Tyne-Hsien Chang
Academic Rank:	Associate Professor
University & Department:	Texas A&M University at Galveston Maritime System Engineering Galveston, TX

NASA/JSC

Directorate:	Engineering
Division:	Navigation Control and Aeronautics
Branch:	Aeroscience
JSC Colleague:	C.P. Li, Ph.D.
Date Submitted:	August 10, 1993
Contract Number:	NGT-44-001-800

ABSTRACT

An unified approach for solving both compressible and incompressible flows has been investigated in this study. The difference in CFD code development between incompressible and compressible flows is due to the mathematical characteristics. However, if one can modify the continuity equation for incompressible flows by introducing pseudocompressibility, the governing equations for incompressible flows would have the same mathematical characters as compressible flows. The application of a compressible flow code to solve incompressible flows becomes feasible. Among numerical algorithms developed for compressible flows, the Centered Total Variation Diminishing (CTVD) schemes possess better mathematical properties to damp out the spurious oscillations while providing high-order accuracy for high speed flows. It leads us to believe that CTVD schemes can equally well apply to solve incompressible flows.

In this study, the governing equations for incompressible flows include continuity equation and momentum equations. The continuity equation is modified by adding a time-derivative of the pressure term containing the artificial compressibility. The modified continuity equation together with the unsteady momentum equations forms a hyperbolic-parabolic type of time-dependent system of equations. Thus, the CTVD schemes can be implemented. In addition, the boundary conditions including physical and numerical boundary conditions must be properly specified to obtain accurate solution.

The CFD code for this research is currently in progress. Flow past a circular cylinder will be used for numerical experiments to determine the accuracy and efficiency of the code before applying this code to more specific applications.

INTRODUCTION

GOVERNING EQUATIONS

The two-dimensional incompressible-flow equations are as follows:

$$\frac{\partial u}{\partial x}+\frac{\partial v}{\partial y}=0 \tag{1}$$

$$\rho\left(\frac{\partial u}{\partial t}+\frac{\partial u^2}{\partial x}+\frac{\partial uv}{\partial y}\right)=-\frac{\partial P}{\partial x}+\mu\left(\frac{\partial^2 u}{\partial x^2}+\frac{\partial^2 u}{\partial y^2}\right) \tag{2}$$

$$\rho\left(\frac{\partial v}{\partial t}+\frac{\partial uv}{\partial x}+\frac{\partial v^2}{\partial y}\right)=-\frac{\partial p}{\partial y}+\mu\left(\frac{\partial^2 v}{\partial x^2}+\frac{\partial^2 v}{\partial y^2}\right) \tag{3}$$

where u and v are velocity components in the x and y directions, respectively. P is the static pressure. μ denotes the dynamic viscosity. The incompressible governing equations (1) through (3) are mathematically classified as elliptic partial differential equations while compressible governing equations are hyperbolic partial differential equations. Because of the mathematical difference between the hyperbolic and elliptic partial differential equations, the well-developed numerical schemes for compressible flows can not apply directly to solve incompressible flows. However, if one modifies the continuity equation given in equation (1) by introducing artificial compressibility, the resulting incompressible governing equations are of hyperbolic type. The modified continuity equation is given in equation (4).

$$\frac{\partial p}{\partial t}+\frac{\partial \beta u}{\partial x}+\frac{\partial \beta v}{\partial y}=0 \tag{4}$$

where β is known as the pseudocompressibility constant.

Equations (2) through (4) are transformed into generalized curvilinear coordinates, ξ and η given by

$$\xi = \xi\,(x, y, t) \tag{5}$$

$$\eta = \eta\,(x, y, t) \tag{6}$$

The governing equations are then given by

$$\frac{\partial D}{\partial \tau} + \frac{\partial (E - E_v)}{\partial \xi} + \frac{\partial (F - F_v)}{\partial \eta} = 0 \tag{7}$$

where

$$D = \frac{1}{J}\begin{bmatrix} P \\ \rho u \\ \rho v \end{bmatrix} \tag{8}$$

$$E = \frac{1}{J}\begin{bmatrix} \beta\,(U - \xi_t) \\ \rho u U + \xi_x P \\ \rho v U + \xi_y P \end{bmatrix} \tag{9}$$

$$F = \frac{1}{J}\begin{bmatrix} \beta\,(V - \eta_t) \\ \rho u V + \eta_x P \\ \rho v V + \eta_y P \end{bmatrix} \tag{10}$$

$$F_v = \frac{\mu}{J}\begin{bmatrix} 0 \\ (\xi_x^2 + \xi_y^2) u_\xi + (\xi_x \eta_x + \xi_y \eta_y) u_\eta \\ (\xi_x^2 + \xi_y^2) v_\xi + (\xi_x \eta_x + \xi_y \eta_y) v_\eta \end{bmatrix} \tag{11}$$

$$E_v = \frac{\mu}{J}\begin{bmatrix} 0 \\ (\xi_x\eta_x+\xi_y\eta_y)u_\xi + (\eta_x^2+\eta_y^2)u_\eta \\ (\xi_x\eta_x+\xi_y\eta_y)v_\xi + (\eta_x^2+\eta_y^2)v_\eta \end{bmatrix} \tag{12}$$

U and V are contravariant velocities. Consequently, a chosen numerical scheme developed for compressible flows can be used to solve incompressible flows.

NUMERICAL SCHEME

Numerical algorithms recently developed for high-speed flows have demonstrated the superiority in reducing CPU time while providing the high-order accuracy. The numerical algorithms are based on the finite volume approach. A fourth-order centered scheme (CTVD) developed by Sanders and Li is used to approximate spatial derivatives in the resulting hyperbolic equations. The CTVD differencing proposed in this study has distinctive desired properties in overcoming the spurious oscillations and odd- and even-point decoupling in the solution which are caused by the use of central differencing. In addition, the implementation of these algorithms is simple and no tuning parameters are needed.

By applying the centered differencing to the transformed equations a system of time dependent differential equations is obtained. The system of differential equations can be integrated by a number of methods to obtain a converged solution. In this study, the Runge-Kutta and ADI methods are used in the code development, respectively.

When the algorithms are applied to solve the incompressible flows, a set of corresponding boundary conditions must be specified including analytical boundary conditions and numerical boundary conditions as well. It is crucial to the accuracy of the solution.

The development of numerical code based on CTVD is still in progress. It will take about one year to complete the code development. Numerical experiments will follow when the code is completed. Two-dimensional problems will be investigated first to study the accuracy and efficiency of the numerical algorithms. Then the code will be extensively used to solve three-dimensional incompressible flow problems with application to bay and nearshore circulation, and others.

CONCLUDING REMARKS

The research to develop an accurate and efficient CFD code for incompressible flows using the algorithms that are originally developed for compressible flows is a viable approach. The significant achievements for compressible flows in code development have been providing a valuable CFD tool for those who are interested in study phenomena of incompressible flows.